SCT

SPECIALTY END MILL

Helical Chamfer Mill

Corner Rounding

Engraving Tool

SPECIALTY END MILL PRODUCT OVERVIEW

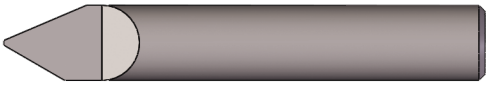
Specialty end mills feature the same premium submicron carbide as the rest of the product lines. They are ground on modern CNC tool-and-cutter grinders to tight tolerances and have been engineered for high performance.

Helical Chamfer Mills (p.121)

Helical Chamfer Mills are made to mill a chamfer on an edge. They come with either three or five helical flutes. The diameter sizes range from 1/8" to ¾", and have included angles of 60, 90, and 120 degrees. The tools are not recommended for plunging countersinks.

Corner Rounding End Mills (p.122)

Corner Rounding End Mills have three flutes and are double ended to provide maximum value. The cutter diameter and the cut depth are held to ±0.001 inch tolerance to provide ease of set-up.



Engraving Tools (p.123)

Engraving Tools come in a large variety of angles and sizes. These solid carbide tools will engrave on a large variety of materials. The tool tip is held to ±0.001 inch tolerance for uniformity.

New Product 2020

Spotting Drills

Spotting drills are a valuable first step in the holemaking process. The tool is solid carbide with a precise point angle that is held to a one-degree tolerance for true centering. Covering a wide range of sizes, it is available in 82, 90, 100, 120 and 142 degree point angles.

New Product 2021

Drill Mills

Drill Mills are designed for milling, chamfering, and light spotting applications. They come with two or four helical flutes each offered in included angles of 90 or 120 degrees. The cutter diameter sizes range from 1/8" to ½".

SPECIALTY END MILL TECHNICAL INFORMATION PAGES 124-125

SPECIALTY TOOL - HELICAL CHAMFER MILL TECHNICAL INFORMATION

MATERIAL	ROCKWELL HARDNESS	SPEED (SFM) UNCOATED	SPEED (SFM) AlTiN+	FEED (Inches per tooth) CALCULATED CUTTING DIAMETER							
				<0.125	.125-.1875	.1875-.250	.250-.3125	.3125-.375	.375-.500	.500-.625	.625-.750
Gray Cast Iron	85Rb	250	450	0.0012	0.0022	0.0035	0.0045	0.0050	0.0055	0.0070	0.0090
Ductile Cast Iron	85Rb	180	375	0.0007	0.0015	0.0020	0.0028	0.0035	0.0040	0.0055	0.0070
Carbon Steel	18Rc	225	450	0.0007	0.0015	0.0022	0.0028	0.0035	0.0045	0.0055	0.0070
Alloy Steel	20Rc	200	400	0.0006	0.0012	0.0020	0.0025	0.0030	0.0040	0.0050	0.0060
Heat Treated Alloys	40Rc	100	200	0.0003	0.0007	0.0010	0.0012	0.0018	0.0020	0.0028	0.0035
Tool Steel	20Rc	150	325	0.0006	0.0010	0.0018	0.0022	0.0028	0.0035	0.0045	0.0055
300 Stainless Steel	80Rb	120	250	0.0005	0.0009	0.0015	0.0018	0.0022	0.0028	0.0035	0.0045
400 Stainless Steel	95Rb	140	325	0.0004	0.0009	0.0012	0.0018	0.0022	0.0025	0.0035	0.0045
Nickel Alloy	20Rc	120	175	0.0005	0.0009	0.0012	0.0018	0.0022	0.0028	0.0035	0.0045
Cobalt Alloy	20Rc	140	225	0.0003	0.0006	0.0009	0.0012	0.0015	0.0018	0.0022	0.0030
Titanium	25Rc	160	250	0.0005	0.0009	0.0012	0.0018	0.0022	0.0028	0.0035	0.0045
Aluminum	60Rb	1000	1900	0.0010	0.0028	0.0040	0.0055	0.0070	0.0080	0.0110	0.0130
Brass, Zinc, Copper	41Rb	320	600	0.0008	0.0015	0.0022	0.0030	0.0040	0.0045	0.0060	0.0080

Determining the Calculated Cutting Diameter

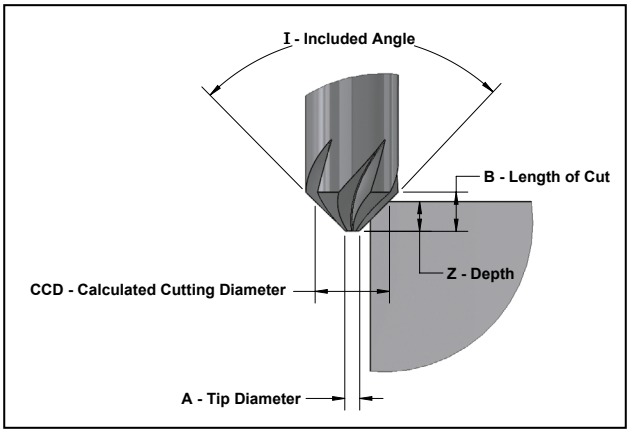
Surface footage and chip load should be calculated at the "Calculated Cutting Diameter" or CCD. The CCD is the largest diameter of the tool that engages the part.

Calculated Cutting Diameter=
2×Depth×tan(Included Angle/2)+Tip Diameter

$CCD=2\times Z\times\tan(I/2)+A$

Choose a "Z – Depth" based on the "B – Length of Cut" of the selected tool. This should result in the part only contacting the included angle of the tool. The part should never touch the shank or tip of the tool. Using a Z – Depth that results in a larger CCD (closer to the shank) is preferred over a smaller CCD (closer to the tip). Find tool dimensions in chart on the product page.

RPM and IPM should be calculated using the Calculated Cutting Diameter.



Example:
Tool: HC50003-090A
I – Included Angle: 90°
A – Tip Diameter: 0.080"
B – Length of Cut: 0.210"
Chosen Z – Depth: 0.200"

Calculation:
$CCD=2\times Z\times\tan(I/2)+A$
$CCD=2\times0.200"\times\tan(90°/2)+0.080"$
$CCD=0.480"$

Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295

SPECIALTY TOOL - CORNER ROUNDING END MILL
FEED AND SPEED CHART

MATERIAL	ROCKWELL HARDNESS	SPEED (SFM) UNCOATED	SPEED (SFM) AlTiN+	FEED (Inches per tooth)				
				Tool Size				
				CR125	CR187	CR250	CR375	CR500
Cast Iron	85Rb	200	350	0.0004	0.0005	0.0009	0.0015	0.0017
Carbon Steel	18Rc	200	400	0.0004	0.0005	0.0008	0.0013	0.0015
Alloy Steel	20Rc	160	330	0.0003	0.0004	0.0007	0.0012	0.0014
Heat Treated Alloys (38-45Rc)	40Rc	60	100	0.0002	0.0003	0.0004	0.0007	0.0009
Tool Steel	20Rc	125	150	0.0002	0.0003	0.0004	0.0007	0.0009
Stainless Steel	95Rb	120	250	0.0003	0.0004	0.0007	0.0012	0.0014
Titanium	25Rc	110	150	0.0002	0.0003	0.0004	0.0007	0.0009
Aluminum	60Rb	450	750	0.0008	0.0011	0.0017	0.0028	0.0033
Brass, Zinc,	41Rb	300	500	0.0007	0.0010	0.0015	0.0025	0.0030

SPECIALTY TOOL - ENGRAVING TOOL
FEED AND SPEED CHART

MATERIAL	RPM	FEED (Inches per tooth)									
		INCLUDED ANGLE									
		30°		40°		60°		90°		120°	
		SHANK DIAMETER									
		.125-.187	.250-.500	.125-.187	.250-.500	.125-.187	.250-.500	.125-.187	.250-.500	.125-.187	.250-.500
Cast Iron	6000+	0.0011	0.0014	0.0012	0.0016	0.0016	0.0020	0.0017	0.0022	0.0019	0.0024
Carbon Steel	6000+	0.0006	0.0008	0.0007	0.0009	0.0009	0.0012	0.0010	0.0013	0.0011	0.0014
Alloy Steel	6000+	0.0005	0.0006	0.0005	0.0007	0.0007	0.0009	0.0007	0.0009	0.0008	0.0010
Heat Treated Alloys	6000+	0.0002	0.0003	0.0003	0.0004	0.0004	0.0005	0.0004	0.0005	0.0004	0.0006
Tool Steel	6000+	0.0004	0.0005	0.0005	0.0006	0.0006	0.0008	0.0007	0.0008	0.0007	0.0009
Stainless Steel	6000+	0.0005	0.0007	0.0006	0.0008	0.0008	0.0010	0.0008	0.0011	0.0009	0.0012
Titanium	6000+	0.0005	0.0007	0.0006	0.0008	0.0008	0.0010	0.0008	0.0011	0.0009	0.0012
Aluminum	6000+	0.0011	0.0014	0.0012	0.0016	0.0016	0.0020	0.0017	0.0022	0.0019	0.0024
Plastics	6000+	0.0016	0.0021	0.0019	0.0024	0.0024	0.0030	0.0026	0.0033	0.0028	0.0036

Suggested chip loads reflect engraving depths up to .010". For depths of cut between .010"-.015", reduce suggested chip loads by 20%. For depths of cut between .015"-.020", reduce suggested chip load by 30%.

Ramping into the part is preferred but if plunge milling into the part, reduce suggested chip load by 50%.